Int. j. adv. multidisc. res. stud. 2025; 5(6):1368-1374

Received: 28-10-2025
Accepted: 08-12-2025

International Journal of Advanced Multidisciplinary Research and Studies

ISSN: 2583-049X

CFD-Based Thermal Optimization of a High-Efficiency Boiler for Paddy Rice Parboiling in Rural Agro-Processing Systems

[1] Awua JT, [2] Ibrahim JS, [3] Agada A, [4] Oche I

[1, 2, 3, 4] Department of Mechanical Engineering, Joseph Sarwuan Tarka University, Makurdi, Nigeria

Corresponding Author: **Awua JT**

Abstract

Parboiling remains a critical pre-processing stage in rice production, especially in Nigeria, where traditional methods are inefficient, labour-intensive, and often result in poor rice quality. This study presents the design and computational fluid dynamics (CFD) analysis of a novel rice parboiling boiler intended to improve energy efficiency and product quality for small-scale rural processors. Using PTC Creo for 3D modelling and ANSYS Fluent 2021 R1 for simulation, the thermal and fluid dynamic performance of the proposed boiler was evaluated under steady-state conditions. Tetrahedral meshing with boundary layer refinement was employed to resolve critical flow gradients, while a realizable k–ε turbulence model was adopted to capture complex internal flow characteristics. Simulation results revealed improved temperature uniformity, smoother fluid flow, and enhanced heat transfer due to the presence of internal baffles. Key performance parameters—including temperature distribution, pressure profile, turbulence kinetic energy, enthalpy, and mass flux—were analysed to validate the design's effectiveness. The boiler achieved a thermal efficiency of 89.95%, demonstrating significant improvements over conventional parboiling systems. This research fills a notable gap in the application of CFD for rice parboiling boiler optimization, offering a cost-effective and energy-efficient solution tailored for rural deployment in developing countries. The findings provide a foundation for future design enhancements and sustainable agro-processing technologies.

Keywords: Rural Agro-Processing, Computational Fluid Dynamics (CFD), Rice Parboiling, Boiler Design, Thermal Efficiency, Heat Transfer, ANSYS Fluent

1. Introduction

Parboiling is a hydrothermal treatment applied to raw paddy and remains a widespread traditional practice in rice processing across Nigeria. Typically, rural farmers—the primary producers of rice—use rudimentary methods, which involve soaking paddy in cold water overnight in mud pots or metal drums for two to three days. This is followed by extended steaming, drying, and milling. However, this conventional parboiling technique often leads to suboptimal results, including incomplete gelatinization, discoloration, and poor market acceptability of the milled rice, primarily due to inconsistent heating and inefficiencies inherent in the process (Umogbai *et al*., 2013) [1]. Moreover, the traditional method is labour-intensive and time-consuming. These limitations underscore the need to design an improved paddy rice boiler that enhances parboiling efficiency, reduces processing time, and ensures better rice quality with greater market value.

Boilers—sealed vessels used to heat fluids, particularly water for rice parboiling—are inherently complex systems. The intricacies of fluid flow and heat transfer within such systems are difficult to analysed through experimental methods alone, given the variety of operating conditions and internal geometries. As a result, numerical simulation techniques, particularly Computational Fluid Dynamics (CFD), have emerged as effective tools for investigating these complex thermal systems (Souhir *et al*., 2018) [2].

CFD enables detailed numerical modelling of fluid dynamics and thermal behaviour within boilers. This approach facilitates the analysis of turbulent flow patterns and complex heat transfer mechanisms throughout the entire boiler volume. Furthermore, CFD can be used to optimize design parameters and operational conditions, ultimately improving combustion efficiency, enhancing heat transfer, and minimizing energy losses (Souhir *et al*., 2018) [2].

Numerous studies have utilized CFD to enhance boiler performance. For instance, Sai Krishna *et al*. (2018) [3] performed thermal and CFD analyses of steam boilers—with and without baffles—used in small power plants. Using CREO for 3D modelling and ANSYS for simulation, they investigated various inlet velocities (20, 30, 40, and 50 m/s) to evaluate heat transfer coefficients, temperature distributions, and pressure drops.

Similarly, Adasu and Ramesh (2018) [4] examined the fluid flow and heat transfer characteristics of an economizer in a tangential-fired tube boiler. Their CFD simulations explored the effects of varying mass flow rates and material properties on thermal performance.

Other notable studies include Mukesh *et al*. (2021) [5], who conducted CFD analysis of boiler tubes to assess temperature, velocity, and pressure distribution; and Ajay *et al*. (2012) [6], who investigated causes of boiler tube leakage through superheater simulations. Deepak *et al*. (2018) [7] used CFD to predict metal temperatures in reheater tubes of a supercritical boiler, identifying potential rupture zones based on thermal stress analysis.

Beesam *et al*. (2018) [8] analysed the thermal performance of steam boilers under different input velocities using SOLIDWORKS for modelling and ANSYS for simulation, while Silva *et al*. (2019) [9] evaluated the impact of air leakage on boiler performance, combustion behaviour, and pollutant formation using Ansys CFX.

Comprehensive reviews, such as that by Shobhit and Mandloi (2020) [10], have highlighted the role of internal geometries—like rifled or corrugated tubes—in enhancing turbulence and, consequently, heat transfer efficiency. Other researchers, including Rajendra *et al*. (2018) [11], have explored innovative design features such as turbulators to improve convective heat transfer and boiler compactness.

Advanced simulation frameworks have also been developed to reflect real-world operational dynamics. For example, Aqeel *et al*. (2018) [12] presented a dynamic simulation of industrial boilers incorporating safety interlocks, start-up/shutdown logic, and fault response validation. Similarly, Praveen (2021) [13] studied air preheater performance using Fluent 14.0 to assess the impact of air leakage and high ash content on heat transfer.

Additional studies have addressed specific boiler types and conditions. Rao and Raju (2018) [14] modelled a bagasse-fired boiler to predict flame profiles and final steam temperatures. Mariusz *et al*. (2019) [15] explored transient-state CFD analysis of superheaters under attemperator operation. Aggala and Mohan (2018) [16] focused on waste heat boilers, evaluating heat recovery effectiveness under different mass flow conditions.

Collectively, these studies underscore the versatility and accuracy of CFD in simulating fluid flow and thermal characteristics of boilers. By leveraging CFD tools, engineers and researchers can optimize boiler design and operation, reduce emissions, improve energy efficiency, and enhance system reliability.

Despite the extensive application of CFD in power and industrial boilers, there remains a significant research gap in its use for analysing and optimizing rice parboiling boilers, particularly those tailored for rural agro-processing applications in developing countries like Nigeria. Most traditional parboiling systems used by local farmers have not been subjected to rigorous thermal-fluid analysis, which limits opportunities for efficiency enhancement and design innovation. This study addresses that gap by applying CFD modelling to analyse the thermal and flow behaviour within a rice parboiling boiler configuration. The novelty of this work lies in its targeted focus on simulating rice parboiling boiler performance using a validated CFD framework, aiming to optimize heat distribution and fluid dynamics for improved parboiling efficiency and rice quality. To the best of our knowledge, this is one of the first studies to use CFD techniques to specifically model and evaluate a rice parboiling boiler system intended for rural deployment, offering new insights that can directly inform the design of more energy-efficient, farmer-friendly parboiling technologies.

2. Materials and Methods

2.1 Computational Setup

The simulation and modelling in this study were conducted using ANSYS Fluent 2021 R1 and PTC Creo Parametric 7.0 for CAD development. The computational analysis was executed on an HP EliteBook 840 G3 laptop equipped with an Intel Core i7 processor, 16 GB RAM, and a 500 GB solid-state drive (SSD).

2.2 Geometry and Meshing

2.2.1 Geometry

3D model representing rice parboiling boiler—combustion chamber, baffle, inlet/outlet manifolds—built in Creo and exported as IGES.

The boiler structure was modelled using PTC Creo Parametric 7.0, a sketch-based, feature-driven, and parametric 3D CAD software developed by PTC. It supports parent–child relationships and bidirectional associativity. Key modelling tools such as extrude, revolve, sweep, and pattern were utilized in developing the geometry. The completed model was then exported in Initial Graphics Exchange Specification (IGES) format for further analysis in ANSYS. Fig 1 presents the CAD representation of the proposed boiler design.

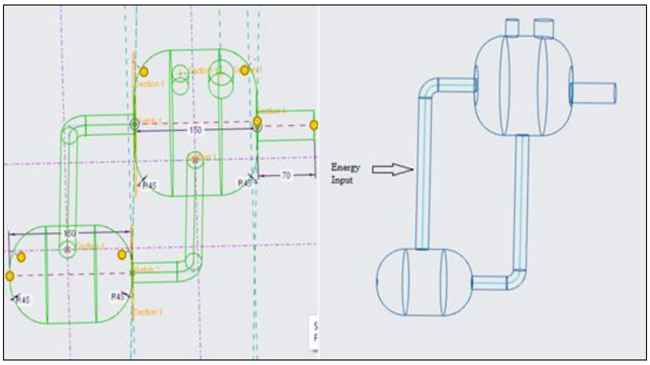

Fig 1: Design of the Proposed Boiler

2.2.2 Mesh Strategy

After completing the boiler's geometry, a tetrahedral mesh was generated using ANSYS ICEM CFD due to its suitability for complex geometries and its efficient element-to-node ratio (~5:1), which reduces memory usage compared to hexahedral meshes (ANSYS ICEM CFD 2021). Fig 2 shows the generated mesh. Post-meshing, quality issues were identified and addressed using ANSYS ICEM CFD's manual and automatic editing tools, including element type conversion, mesh refinement/coarsening, and smoothing.

- Employed tetrahedral meshing for complex internal geometry.
- Boundary-layer refinement applied near walls using prism layers: 10 layers, first thickness ~0.015 mm,

growth ratio ~1.15—mirroring the approach of sensors boiler studies to resolve thermal gradients (Paz *et al.*, 2022) [17].
- Cell size kept ~0.5 mm to avoid skewness and ensure stability (Alpha *et al.*, 2024 and Paz *et al.*, 2022) [18, 17].
- Mesh quality metrics targeted:
- Skewness < 0.9 (max)
- Average skewness < 0.2
- Mesh independence verified: incremental refinement ceased when deviation in outlet temperature/pressure < 1%.

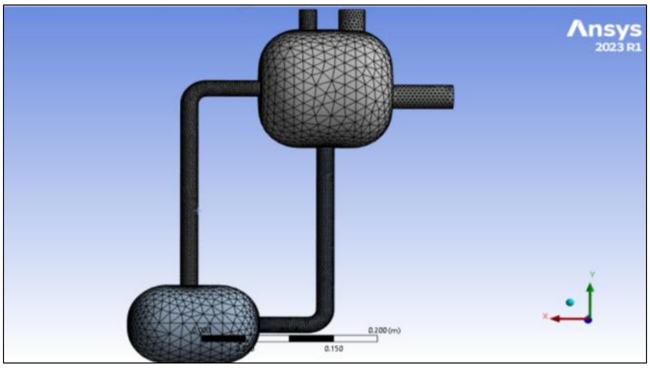

Fig 2: Meshing for the Proposed Boiler

2.3 Physical Models & Boundary Conditions

Accurate boundary conditions are essential for reliable CFD simulations. In this study:
- Flow Model: Steady-state, 3D ideal-gas single-phase flow.
- Turbulence: Realizable *k–ε* model employed, consistent with best practices for turbulent boiler flow (Tang *et al.*, 2022) [19].
- Combustion: Energy equation enabled with volumetric heat source mimicking actual baffle-generated heat.
- Radiation: Neglected due to significance primarily in large-scale or oxy-fuel systems.
- Heat Flux: Wall heat flux fixed at 200 W/m².
- Inlet velocity condition were varied as shown in Table 1

Table 1: Boundary conditions summary

Domain Item	Specification
Inlet	Uniform velocity entry (0.1– 0.5 m/s)
Outlet	Pressure-outlet boundary
Walls	No-slip, constant heat flux
Symmetry axis	Axisymmetric boundary condition

2.4 Solver Settings & Convergence

- **Solver:** Pressure-based, with second-order discretization for momentum and energy.
- **Pressure–Velocity Coupling:** *SIMPLE* algorithm, selected for its stability in steady-state recirculating systems (Tang *et al.*, 2023; Alpha *et al.*, 2024) [20, 18].
- **Upwind Scheme:** Second-order upwind for all transport equations.
- **Under-Relaxation Factors:** Momentum 0.3, Pressure 0.1, Energy 0.8 to avoid divergence.

Convergence Criteria:

- Residuals below 1×10^{-6} for continuity and momentum, 1×10^{-8} for energy.
- Monitoring of outlet mass flow and max temperature; convergence confirmed when changes < 0.1% over 500 iterations.

2.5 Grid Independence and Sensitivity

- Tested at baseline mesh and two refinement levels.
- Temperature/pressure variation at outlet between baseline and finest mesh was <1%, confirming grid independence.

2.6 Post-Processing

- Extracted contour plots of temperature, pressure, TKE, enthalpy, and mass flux.
- Analysed along central plane and baffle-adjacent regions to evaluate mixing efficiency and design improvements.

2.7 Thermal Efficiency Calculation

The boiler efficiency was determined by employing the Carnot efficiency formula, which is given by:

$$\text{Efficiency} = 1 - \frac{T_{low}}{T_{high}} \quad (1)$$

Where T_{high} and T_{low} are the absolute highest and lowest temperature respectively.

2.8 Validation & Best Practices

Methodology draws on validated boiler-CFD conventions: layered meshing, *k–ε* turbulence, SIMPLE coupling, and rigorous grid convergence (Paz *et al.*, 2022; Silva *et al.*, 2019) [17, 9].

As recommended in sensor-fluidized bed literature, model simplicity was increased first to isolate effects before including full geometry with baffle (Adamczyk 2017) [21]. Table 2 shows summary of the computational fluid dynamic procedure.

Table 2: Summary Table of CFD Procedure

Step	Description
Geometry creation	PTC Creo → IGES
Meshing	Tetrahedral + prism layers near walls
Solver setup	Pressure-based, *k–ε*, heat source, SIMPLE coupling
Discretization schemes	Second-order upwind for momentum & energy
Boundary conditions	Inlet velocity sweep, outlet pressure, fixed wall flux
Convergence criteria	Residuals, outlet monitoring, grid independence
Post-processing	Contour plots + efficiency calculation

3. Results and Discussion

3.1 Temperature Distribution

Fig 3 presents the three-dimensional temperature distribution within the proposed boiler configuration. The results indicate that the modified design significantly influences the temperature profile throughout the boiler volume. Notably, the highest temperature values were recorded in the proposed design, suggesting enhanced thermal performance. This improvement is attributed to the incorporation of a baffle, which effectively alters the internal flow dynamics and combustion characteristics.

The presence of the baffle promotes better mixing of the combustion gases, resulting in more complete combustion and elevated temperature levels within the combustion chamber. This rise in temperature enhances heat transfer efficiency, thereby increasing the conversion of fuel energy into useful thermal energy. Such improvements are essential for reducing fuel consumption and operational costs (Souhir *et al*., 2018; Mekala and Nagaraju, 2018) [2, 22].
Furthermore, the elevated temperatures contribute to a reduction in the moisture content of the generated steam and lower the risk of condensation-induced corrosion. These advantages underscore the effectiveness of the proposed design modifications in improving boiler performance and durability.

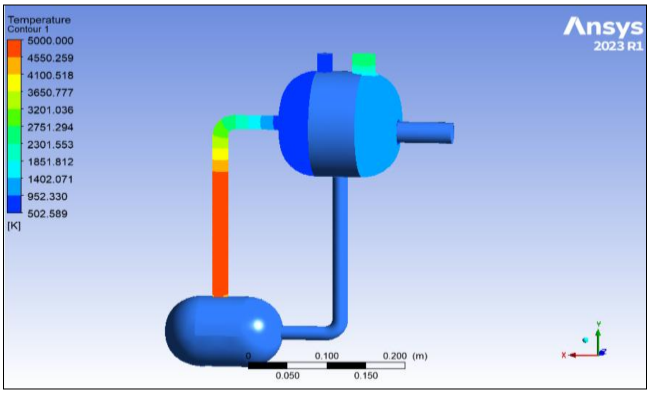

Fig 3: 3-D Temperature Distribution Contour for the Proposed Boiler

3.2 Pressure Distribution

Fig 4 illustrates the pressure distribution within the proposed boiler configuration. The simulation results reveal a distinct pressure pattern, characterized by an initial increase in pressure within the combustion chamber, followed by a gradual decline toward the boiler outlet. Notably, regions of low pressure are observed both within the boiler core and at the outlet section.
This pressure profile is indicative of enhanced thermal and fluid dynamic performance, contributing significantly to the overall efficiency of the boiler system. The presence of lower pressure zones implies reduced mechanical stress on structural components, thereby minimizing the risk of rupture or leakage, as supported by Souhir *et al*. (2018) [2]. Moreover, operating under lower pressure conditions translates to decreased energy demand for fluid circulation, ultimately leading to lower power consumption and reduced operational costs.

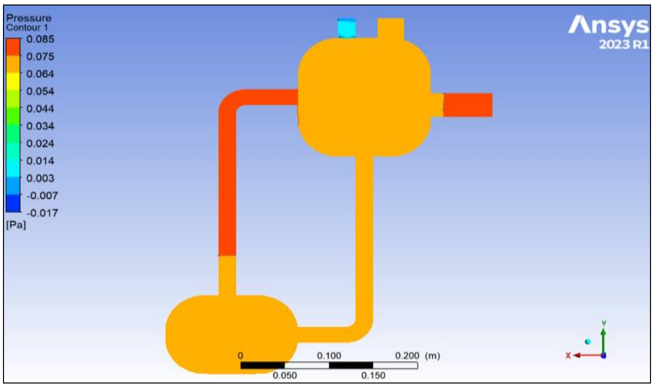

Fig 4: 3-D Pressure Distribution Contour for the Proposed Boiler

3.3 Turbulence Kinetic Energy Distribution

Fig 5 shows the turbulent kinetic energy distribution inside the modified boiler. The design of the proposed boiler influences the turbulent kinetic energy distribution inside the boiler. The turbulence kinetic energy increases significantly under the boiler outlet. At the outlet, it reaches its maximum value. In fact, it emerges maximal under the whole outlet of the boiler. The lower turbulence kinetic energy recorded in the proposed boiler means that there is a smoother flow of fluid in the boiler, and this enhances the efficiency of the boiler, because turbulence causes energy losses by increasing the resistance to fluid flow in the boiler. Excessive turbulence hinders heat transfer efficiency in boiler by disrupting boundary layers and promoting heat loss (Souhir *et al.,* 2018) [2]. Lower turbulence in the case of the proposed boiler facilitates better heat transfer, improving the overall performance of the boiler.

3.4 Turbulence Kinetic Energy Distribution

Fig 5 presents the distribution of turbulence kinetic energy (TKE) within the modified boiler. The geometry of the proposed design significantly influences the internal turbulence characteristics. A notable increase in TKE is observed near the boiler outlet, where it attains its peak values, indicating intensified flow disturbances in this region. However, the overall turbulence levels within the proposed design remain relatively lower compared to conventional configurations.
This reduction in turbulence is indicative of a smoother and more streamlined flow, which contributes positively to the boiler's thermal and hydraulic performance. Excessive turbulence is known to impede heat transfer by disrupting the thermal boundary layer and increasing flow resistance, thereby leading to energy losses (Souhir *et al*., 2018) [2]. In contrast, the moderated turbulence levels observed in the proposed boiler design support more stable boundary layer development, facilitating efficient heat transfer and enhancing overall boiler performance.

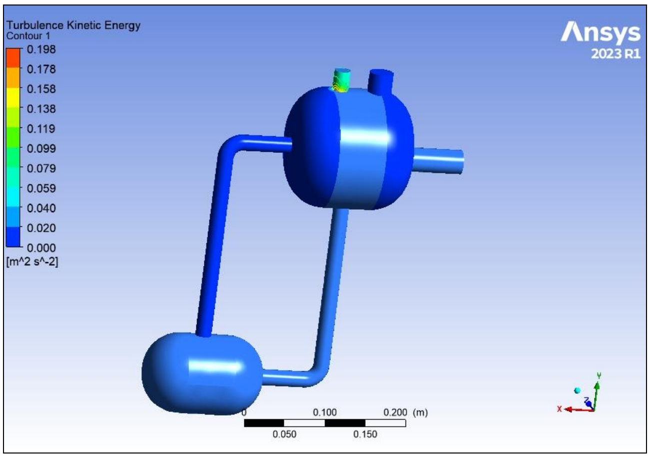

Fig 5: 3-D Turbulence Kinetic Energy Distribution Contour for the Proposed Boiler

3.5 Density Distribution

Fig 6 illustrates the density distribution throughout the entire volume of the boiler. The results reveal a uniform density profile, indicating that the fluid density remains constant across different regions within the boiler. This uniformity plays a critical role in maintaining stable pressure conditions, thereby ensuring safe and reliable boiler operation. Moreover, the consistent density facilitates uniform heat transfer across the system, minimizing the risk of localized overheating or underheating, which could otherwise compromise thermal performance and system integrity (Souhir *et al*., 2018) [2].

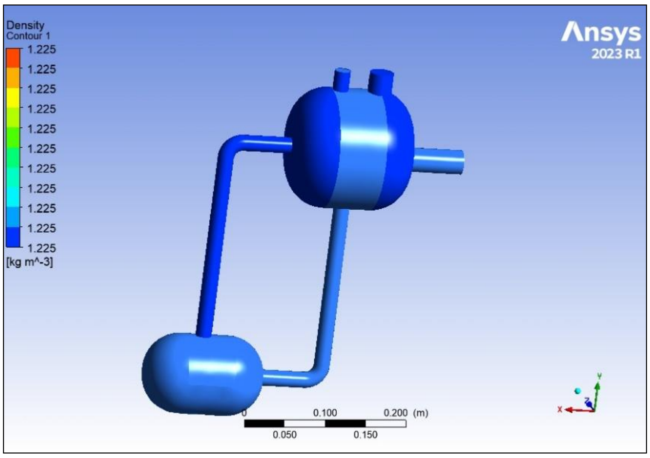

Fig 6: 3-D Density Contour for the Proposed Boiler

3.6 Static Enthalpy

Static enthalpy quantifies the total energy content of the fluid within the boiler and is a critical parameter for assessing thermal performance. As illustrated in Fig 7, the proposed boiler configuration exhibits a higher static enthalpy compared to conventional designs. This indicates enhanced energy availability for heat transfer processes, facilitating more rapid fluid heating and contributing to improved overall thermal efficiency. Additionally, the elevated static enthalpy enables more precise control of temperature fluctuations, ensuring operation within optimal thermal limits for both performance enhancement and operational safety.

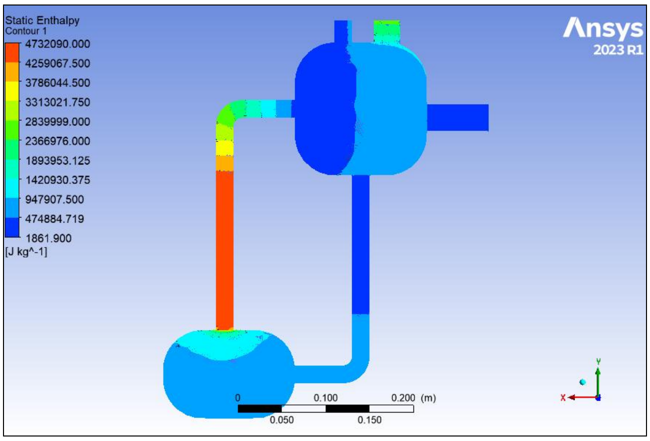

Fig 7: 3-D Static Enthalpy Contour for the Proposed Boiler

3.7 Mass Flux

Mass flux is a fundamental parameter in boiler design and operation, as it quantifies the mass flow rate of the working fluid (water or steam) per unit cross-sectional area of the tube. It directly influences the system's heat transfer efficiency, steam generation capacity, and overall thermal performance. Higher mass flux values typically indicate a greater rate of fluid movement through the boiler tubes, which enhances convective heat transfer and supports improved thermal efficiency.

In the proposed boiler configuration, as shown in Fig 8, a notably higher mass flux was observed compared to conventional designs. This elevated mass flux signifies that a larger volume of fluid flows through the tubes per unit time and surface area, facilitating increased steam production within the same operational period. The result is a higher steam output, which directly contributes to the system's power generation capacity and responsiveness to load demands.

Moreover, the enhanced mass flux promotes more effective heat transfer between the heating surfaces and the circulating fluid. Faster-moving fluid absorbs thermal energy more efficiently from the combustion chamber or heat source, enabling the boiler to achieve higher operating temperatures and pressures while maintaining material integrity and system safety. This improvement supports more stable and controlled steam production, which is critical for downstream components such as turbines and heat exchangers.

Operating at elevated temperatures and pressures enabled by increased mass flux improves the boiler's overall thermal efficiency. The system can extract more usable energy from the fuel, thereby reducing fuel consumption and emissions. Hence, the higher mass flux observed in the proposed boiler design is a key performance enhancement, offering significant advantages in steam generation efficiency, operational stability, and energy output.

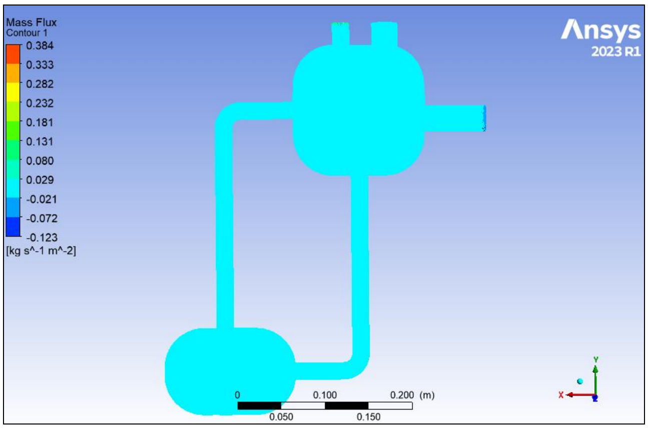

Fig 8: 3-D Mass Flux Contour for the Proposed Boiler

3.8 Boiler Efficiency

The thermal efficiency of the proposed rice parboiling boiler was evaluated based on the ratio of useful energy output to the total energy input, considering the combustion characteristics of the selected fuel and the design parameters of the boiler. The calculated efficiency was found to be **89.95%**, indicating a high level of energy conversion performance. This result demonstrates that the boiler design effectively minimizes heat losses through optimized heat transfer surfaces, appropriate insulation, and efficient combustion processes. Such a high efficiency implies that a significant portion of the energy generated from fuel combustion is utilized in the parboiling process, thereby reducing fuel consumption and operational costs. Moreover, the efficiency value is within the acceptable range for modern biomass-fired boiler systems, underscoring the potential of the proposed design for sustainable and cost-effective rice processing applications, particularly in rural and off-grid communities.

4. Conclusion

This study successfully demonstrated the modelling and simulation of a proposed boiler specifically designed for parboiling paddy rice, addressing the inefficiencies associated with traditional rice processing methods commonly used in rural Nigeria. By employing advanced computational fluid dynamics (CFD) techniques in ANSYS Fluent, the thermal and fluid flow behaviour within the boiler was comprehensively analysed, revealing key performance improvements brought about by the inclusion of a baffle and optimized geometry. The results showed uniform temperature distribution, reduced turbulence, and enhanced mass flux, all contributing to more efficient heat transfer and steam generation. With a calculated thermal efficiency of 89.95%, the proposed design offers a significant step forward in developing energy-efficient, cost-effective, and durable parboiling technologies suitable for rural agro-processing.

The novelty of this work lies in its application of CFD simulation tools to a rice parboiling boiler—a largely understudied area—thus providing a new pathway for improving local rice processing through data-driven design. The insights gained from this simulation can guide the fabrication and field deployment of improved boiler systems, potentially transforming rice parboiling practices and supporting food security and economic empowerment in rural communities.

Future work will focus on experimental validation of the simulation results and the development of a prototype system for field testing under real operating conditions.

5. References

1. Umogbai VI. Development of a Farm Level Paddy Rice Parboiling Device. International Journal of Recent Technology and Engineering. 2013; (2)2:16-21.
2. Souhir E, Abdallah B, Zied D, Mohamed SA. CFD Simulation and Optimization of Industrial Boiler. Energy, 2018. Doi: https://doi.org/10.1016/j.energy.2018.12.006
3. Sai Krishna M, Subbarao E, Kiran K. Design and Thermal Analysis of Steam Boiler (Without & With Baffles) Used in Small Power Plants. International Journal of Advanced Research in Science, Engineering and Technology. 2018; 5(5).
4. Adasu T, Ramesh B. Modelling and Computational Fluid Dynamics Analysis of Economizer in Tangential Fired Tube Boiler. International Journal of Innovative Technology and Research. 2018; 6(6).
5. Mukesh K, Piraisoodan S, Sridhar M, Tamilselvan P, Inbasekaran K. CFD Analysis of Steam Boiler Tube Used in Power Plant. IJARIIE. 2021; (7)4. Om of Boiler, http://Betterbricks.Com/Sites/Default/Files/Operations/Om_Of_Boilers_Final.Pdf
6. Ajayi N Ingale, Vivek CP, Vivek HT. CFD Analysis of Superheater in View of Boiler Tube Leakage. International Journal of Engineering and Innovative Technology. 2012; (1)3.
7. Deepak KK, Ravi KA, Sachin KSH. International Journal of Engineering and Technology. 2018; (10)3. Doi: 10.21817/Ijet/2018/V10i3/181003011
8. Beesam N, Nachiket GC, Prakash N, Madhavi P. Analysis of the Steam Boiler Used in a Power Plant's Thermal and Computational Fluid Dynamics. International Journal of Research and Analytical Reviews. 2018; (5)4.
9. Silva CV, Dondoni LF, Ermel C, Bellani PL, Stradioto DA, Indrusiak MLS. CFD analysis of the combustion process in a boiler of a 160 MWe power plant: Leakage influence. Journal of the Brazilian Society of Mechanical Sciences and Engineering. 2019; 41(7):287. Doi: https://doi.org/10.1007/s40430-019-1787-7
10. Shobhit KG, Mandloi RK. A Literature Review on Computational Fluid Dynamic Analysis of Water Tube Boiler and Heat Exchanger Technique. International Research Journal of Engineering and Technology. 2020; (7)6.
11. Rajendra PA, Vaidyanathan S, Anand C. Design and Analysis of Compact Boiler. International Journal of Mechanical Engineering and Technology. 2018; (9)7:1525-1534.
12. Aqeel AT, Mohammad MA, Muhammad OF, Muhammad ES, Muhammad Z, Waqas S, *et al.* Comprehensive Dynamic Modeling, Simulation, and Validation for an Industrial Boiler Incident Investigation. American Institute of Chemical Engineers, Process Safety Progress, 2018. Doi 10.1002/prs.12040
13. Praveen M. Experimental and Computational Fluid Dynamics Simulation of Performance Analysis of Steam Generators of Boilers. Indian Journal of Power and River Valley Development, 2021.
14. Rao, Ravi Raju. An Anlytical Study in Finding Dynamic Performance of a Steam Boiler by Using CFD. International Journal of Research in Engineering and Applied Sciences. 2018; (3)5.
15. Mariusz G, Marcin T, Jan T. CFD Analysis of Steam Superheater, Transient State. E3s Web of Conferences. 2019; 128:04009. Doi: https://doi.org/10.1051/e3sconf/201912804009
16. Aggala AK, Mohan A. Design and Fluid Flow Analysis of Waste Heat Boiler in CFD. Journal of Emerging Technologies and Innovative Research. 2018; (5)6.
17. Paz C, Suárez E, Cabarcos A, Díaz A. Experimental investigation and cfd analysis of pressure drop in an orc boiler for a whrs implementation. Sensors. 2022; 22(23):9437. Doi: https://doi.org/10.3390/s22239437
18. Alpha NA, Aondover IH, Kuhe A. Experimental and numerical studies of the effect of perforation configuration on heat transfer enhancement of pin fins heat sink. Heat Transfer. 2024; 53(5):2525-2555. Doi: https://doi.org/10.1002/htj.23051
19. Tang Z, Dong H, Zhang C, Cao S, Ouyang T. Deep learning models for SO_2 distribution in a 30 mw boiler via computational fluid dynamics simulation data. ACS Omega. 2022; 7(46):41943-41955. Doi: https://doi.org/10.1021/acsomega.2c03468
20. Tang Z, Yang Y, Pan L, Sui M, Cao S. Fast prediction of heat flux distribution in boilers using computational fluid dynamics simulation data via multi-extreme learning machines. Journal of Chemical Engineering of Japan. 2023; 56(1):2260416. Doi: https://doi.org/10.1080/00219592.2023.2260416
21. Adamczyk WP. Application of the numerical techniques for modelling fluidization process within industrial scale boilers. Archives of Computational Methods in Engineering. 2017; 24(4):669-702. Doi: https://doi.org/10.1007/s11831-016-9186-z

22. Mekala ER, Nagaraju J. Thermal And CFD Analysis for Finding Heat Transfer Through an Angular Bend Tube For Various Nano Fluids. International Journal of Research. 2018; (5)16.